A computational study of the movement of an object driven by the centrifugal pump in the pipeline based on overset meshes

†Jie Chen[1], *JingYin Li[1], Le Wang[2], and JunWei Su[3]

[1]School of Energy And Power Engineering, Xi'an Jiaotong University, China.
[2]School of Civil Engineering, Southwest Jiaotong University, China.
[3]School of Human Settlements and Civil Engineering, Xi'an Jiaotong University, China.

*Presenting author: vrnetgod@qq.com
†Corresponding author: jyli@mail.xjtu.edu.cn

Abstract

The overset mesh method was used for numerical calculation of the movement process of a moving object in a pipe driven by a pump, the appropriate set solving parameters, such as mesh scale, iteration, time step, etc., were determined by reducing the fluctuation of the force applied to the object, the movement process of the object in the pipe from stillness to linearly increased revolution speed was explored, the movement process and movement law of the moving objects with different diameters were analyzed, the change in flow field of each moving object at the gap between it and the pipe wall was compared, and the effect of the gap on the fluid force on the object was analyzed in this paper. The simulation results showed that the forces on and movement fluctuations of moving objects at different gaps were different, and the gap sizes also significantly affected the fluctuations of fluid forces on the moving objects.

Keywords: transient flow; CFD; overset; centrifugal pump

Introduction

The transient operation of a centrifugal pump exists in many occasions. For example, underwater weapon launching is a typical engineering application of the start process of a centrifugal pump. Such process requires the lift and flow of the pump to reach the given indices within a short time. The pump always is starting until it stably works. This start process is significantly different from the stable operation process of this pump. When this centrifugal pump is sued as an underwater launcher, this launching process is equivalent to the movement process of an object in a pipe driven by it. The problem about the transient performance of this centrifugal pump which drives the object in this pipe is different from the previous studies on the start characteristics of a single pump and the flow in the pipe. This problem leads to correlation between start process of this pump and the movement process of the object. Thus, it is necessary to further study the numerical calculation of this transient performance. The proposal of an appropriate numerical calculation method is very important to predict this dynamic process by the start characteristics of this centrifugal pump and numerical calculation of the object movement process. The study result will provide a reference for design and application of the start process of a centrifugal pump.

The study on the transient characteristics during transient start of a centrifugal pump goes all the way back to relevant researchers' theoretical analysis and numerical calculation of water hammer phenomenon in the early 20th century. Allievi[1] proposed a method of solving the water hammer equation. Schnyder and Bergeron[2] proposed a graphic method for analysis of the water hammer phenomenon, respectively. Wylie and Streeter[3] systematically discussed the transient flow mechanism and proposed the theoretical model of solving several transient processes of hydraulic machinery. Akimoto. Narumi[4] defined the physical concepts of water hammer and pressure fluctuation and systematically described the calculation method and the method for prevention of water hammer. In addition, foreign researchers did a lot of experimental studies. For example, Tsukamoto and Tanaka[5]-[7] experimentally studied various transient processes of a centrifugal pump, and used high speed photography to capture the cavitation damage in the transient processes. Domestic researchers did a lot of studies on the transient start characteristics of a pump. Wu Dazhuan[8] used MRF method for numerical simulation of transient start process of one centrifugal pump to obtain the development process of flow field in the transient start process. Wang Leqin et al [9][10] used power bond graph method to build a mathematical model for start process of a vane pump, and proved the applicability of the power bond graph method by comparing with the experimental results. Li Zhifeng et al[11] did a lot of experimental studies on the various transient processes of a vane pump and used the moving mesh method and large eddy simulation technique for unsteady numerical simulation of the start process of a centrifugal pump; their studies showed that one transient pressure shock phenomenon existed in the pump during quick start, and the lift increased with increasing revolution speed; when the revolution speed reached a maximum value and was steady, the transient performance became steady state performance, and the flow increment showed a lag effect in the transient start process. Hu Fangfang et al. [12] used the moving mesh method for numerical simulation in the transient start process of a centrifugal pump, and their simulation results showed that a large eddy zone, which existed in the impeller channel during start, gradually shifted from impeller outlet to the middle of the impeller channel in the start process.

Relevant studies showed that the study on the transient characteristics of a pump by sliding mesh method is relatively mature. The movement of an object in a pipe is usually studied by sliding mesh method, which requires call by value through an interface between pipe wall zone and object zone, and use of dynamic layering method to split and merge the mesh layer at boundary between moving zone and stationary zone according to the change in height of the moving mesh at the boundary. To ensure the calculation accuracy, sliding mesh method requires mesh densification in the pipe wall zone and object zone. A very small gap between the pipe wall and object will greatly increase the number of meshes. In addition, the mesh splitting and merging at the boundary also increase the additional calculation load. The overset mesh method is an important mesh method to process the relative movement of an object [13][14]. In this method, computational mesh is split into many overset or overset submesh. When an object is moving, the body-fitted mesh moves. The numerical calculation is performed on each submesh, and information transfer of the flow field is achieved by interpolation on the overset submesh. It should be noted that the sliding mesh method [15][16] is a special overset mesh method with minimum overset area. The main advantage of the overset mesh method is to reduce the difficulty in mesh generation, enhance the mesh generation flexibility, guarantee initial mesh quality, and inherit the initial solver better. A mesh zoning strategy, the overset mesh method is very favorable for parallel calculation. When the overset zone is moving, the interface for the sliding mesh method need not be artificially defined and the mesh at the interface need be split and merged by dynamic model. Thus, the number of meshes can be reduced and the mesh splitting and merging time can be saved when the overset mesh is used for the object movement in a pipe. However, the overset mesh quality greatly influences the value solving accuracy and convergence rate. The author of the literature [17] proposed the advice on meshblock and hole boundary position, overset zone size, and mesh quality at the overset zone, An overset mesh method suitable for the object movement in a pipe was studied in consideration of relevant advice in this paper.

Calculation Assumption and Physical Model

In the simulation calculation, it is assumed that the revolution speed of a pump is linearly increased to 960 rpm within 5 s, i.e. transient speed, N=960/5*t, and the effect of the gravity on water and moving object is negligible.

Continuity equation:

$$\nabla \cdot \vec{u} = 0 \tag{1.}$$

Momentum equation:

$$\rho \frac{\partial u}{\partial t} + \rho div(u\vec{u}) = \mu \nabla^2 u - \nabla p \tag{2.}$$

$$\rho \frac{\partial v}{\partial t} + \rho div(v\vec{u}) = \mu \nabla^2 v - \nabla p \tag{3.}$$

$$\rho\frac{\partial w}{\partial t}+\rho div(w\vec{u})=\mu\nabla^2 w-\nabla p \quad (4.)$$

Turbulence model: The turbulence model used in this paper is $k-\omega$ SST model. $k-\omega$ model was initially proposed by Wilcdx[19], and its advantage is the processing of near wall in calculation at low Reynolds number. It does not the complicated non linear damping function required in the $k-\varepsilon$ model, and therefore is more accurate and steadier than $k-\varepsilon$ model.
k and ω meet the following transport equations:

$$\frac{D(\rho k)}{Dt}=P_k-\beta^*\rho\omega k+\frac{\partial}{\partial x_j}[(\mu+\sigma^*\mu_T)\frac{\partial k}{\partial x_j}] \quad (5.)$$

$$\frac{D(\rho\omega)}{Dt}=\frac{\gamma\omega P_k}{k}-\beta\rho\omega^2+\frac{\partial}{\partial x_j}[(\mu+\sigma\mu_T)\frac{\partial\omega}{\partial x_j}] \quad (6.)$$

$$\mu_T=\gamma^*\frac{\rho k}{\omega} \quad (7.)$$

Menter believed that the initial $k-\varepsilon$ model was very sensitive to free flow conditions [20]. The calculation results from the model are very different at different ω values at the inlet. To solve this problem, Menter developed a method: $k-\omega$ model was used for the near wall; $k-\varepsilon$ model was used for the outside of the boundary layer, and both the turbulence models are used for the inside of the boundary layer. The $k-\omega$ SST model equation is as follows:

$$\frac{D(\rho k)}{Dt}=P_k-\beta^*\rho\omega k+\frac{\partial}{\partial x_j}[(\mu+\sigma_k\mu_T)\frac{\partial k}{\partial x_j}] \quad (8.)$$

$$\frac{D(\rho\omega)}{Dt}=\frac{\gamma\rho P_k}{\mu_T}-\beta\rho\omega^2+\frac{\partial}{\partial x_j}[(\mu+\sigma\mu_T)\frac{\partial\omega}{\partial x_j}]+2(1-F_1)\frac{\rho\sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j} \quad (9.)$$

$$\mu_T=\frac{a_1\rho k}{\max(a_1\omega, F_2\Omega)} \quad (10.)$$

Where $P_k=\tau_{ij}\frac{\partial u_i}{\partial x_j}$ is turbulence kinetic energy production term. For all the parameters in this equation, see the above literature.

Calculation Model

Geometry Model and Boundary Conditions

IS 65-40-250 centrifugal pump in chemical industry was used in calculation. The appearance dimension of the moving object was shown in Figure 1.The objects had the diameter, D, of 80 mm, 90 mm and 100 mm and were expressed as Object-A, Object-B, and Object-C, respectively. One object was installed in transparent pipe with inner diameter of 120 mm. The pump had two pipes with the inner diameter of 32 mm at its ends. One end is connected to a water tank (During calculation, the change in water level was negligible, and the boundary conditions of a stagnation inlet were used), and the other end was transitionally connected to the transparent pipe with inner diameter of 120 mm. The arrangement and boundary conditions were shown in Figure 2.

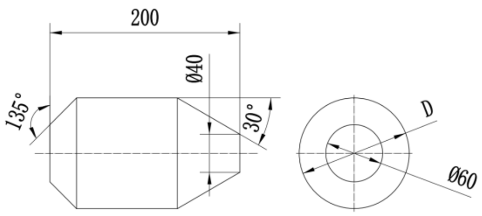


Figure 1. Appearance Dimension of Moving Object

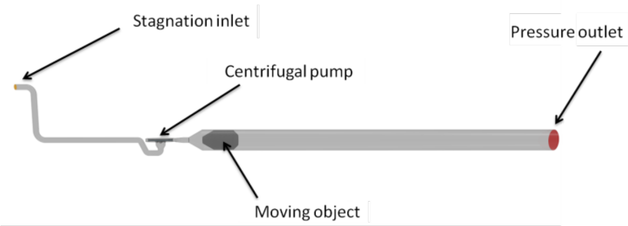
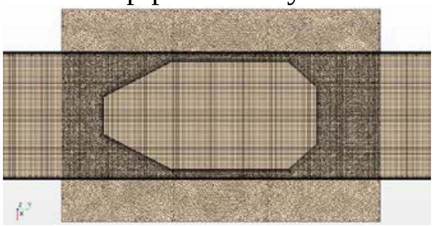


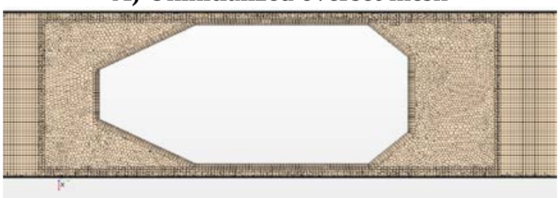
Figure 2. Piping Figure 1. A plate not for meal

Meshing

A computed field mainly is divided into four zones, pump inlet pipe zone, rotary zone, pump outlet pipe zone and moving object zone in the sequence of water flow. In this paper, Trimmer mesh method was used to give pipe zone, rotary zone, and pump outlet pipe zone during numerical calculation. The polyhedral mesh method was used to give a motion zone. Different mesh sizes are used. The calculated forces on the objects were compared. It was found that the forces on the objects fluctuated very sharply, and the force fluctuation was not significantly improved after mesh densification and reduction of time step. Finally, the mesh size was set to 0.002 m considering that a lot of time was spent on calculation. The meshes at each zone generated the boundary layer meshes (Figure 3). Two pipe zones were connected to the rotary zone including pump vane by the interface, while the independent zone including the moving object was connected to the pipe zones by Overset Mesh Interface.



A) Uninitialized overset mesh



B) Initialized overset mesh

Figure 3. Mesh in Computed Field (Object-C)

In the overset mesh method, the calculation was performed by mesh overlapping. This method has such as significant advantage that it can solve the problems of very complicated relative movement at some zones, solid contact in space, etc. However, these problems always are impossibly solved by discontinuous mesh method. The overset mesh mechanism is calculation information transfer between two different zones by overset meshes. Its principle is as follows: In the case of calculation by the overset mesh method, the meshes between two zones (master zone and slave zone) are overset. Based on the interpolation information of the slave zone, the overset zone will re-generate combined overset mesh according to the connection information of the point of the adjacent mesh outside the overset zone as well as the interpolation of the point of the mesh at the overset zone edge and master zone after the calculation begins. In the calculation of the combined overset mesh, the point of the adjacent point outside the overset zone and the point at the overset zone edge are regarded as the boundary points. The values are transferred and exchanged between the mesh at the slave zone and the mesh at the master zone by interpolation. [18] The operation procedure of the overset mesh method mainly consists of five steps: (1) two or more computed fields are created: the background zone covers the flow zone of the far field, and the independent zone surrounds the moving object of concern (overset bodies); (2) all the zones are independently meshed; here would be an overset mesh zone, where the meshes from different zones are overset at one place; (3) the outer boundary of the overset body (overset body) is set to “Overset Mesh”, and “Overset Mesh Interface” is built in the two computed fields; and (4) with moving overset body in the background zone, the overset zone will change; and (5) The information is transferred in two zones by the overset mesh.

Result and Analysis

During calculation, the time step is 0.005s, the six degrees of freedom solver has the number of solving steps of 15 and maximum number of internal circulation steps of 40, and the residual error is set to less than 0.001 in the condition of convergence. A double core (Intel Xeon E5-2680 v4 14C 2.40GHz) workstation is used for calculation, 18 threads parallel calculations of 5s movement takes about 20 days.

The forces on Object-A, Object-B, and Object-C applied by the fluid in a pipe are shown in Figure 4. With decreasing gap between the object and pipe wall, the forces on the moving objects fluctuated very sharply. With increasing revolution speed of the pump, the force fluctuation increases. The fluid forces on three moving objects are in the reverse direction of water flow (positive direction of y-axis) and continuously increase at the time between 2.2s and 2.7s and between 2.7s and 3.2s. The fluid force on Object-A and Object-B sharply reduces at the time of 3.5s and 3.7s, respectively, but that on Object-C does not sharply reduce.

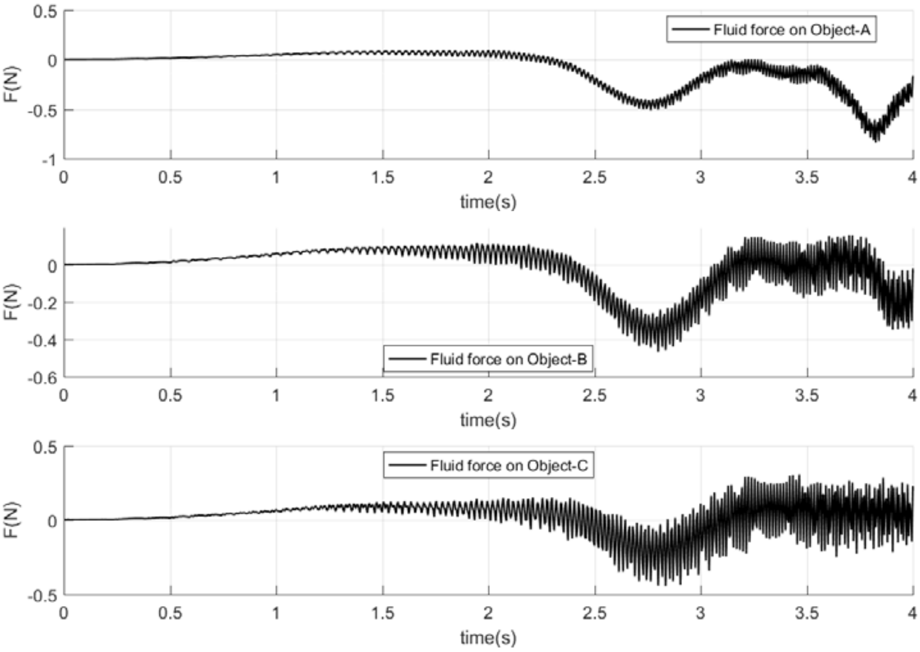


Figure 4. Calculated Force

For different moving objects in the pipe, the fluid force on the pump vane (Figure 5) and torque (Figure 6) change similarly over time, indicating that the loads on the pump vane are not significant different when the pump vane drives the moving objects with different diameters to move mainly because the objects do not significantly affect the flow resistance in the pipe. The change in mass flow (Figure 7) also accounts for this phenomenon.

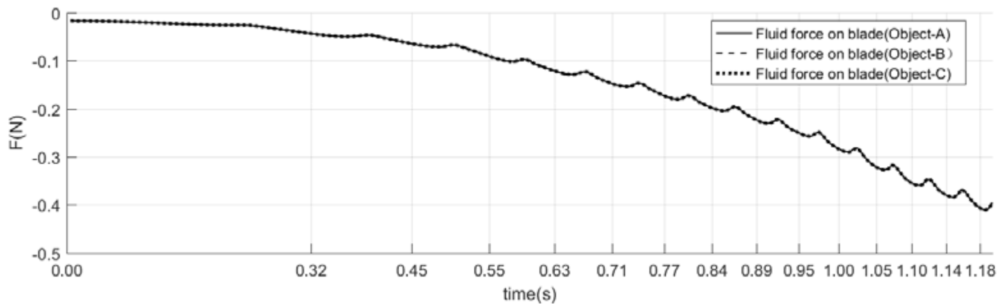


Figure 5. Fluid force on vane

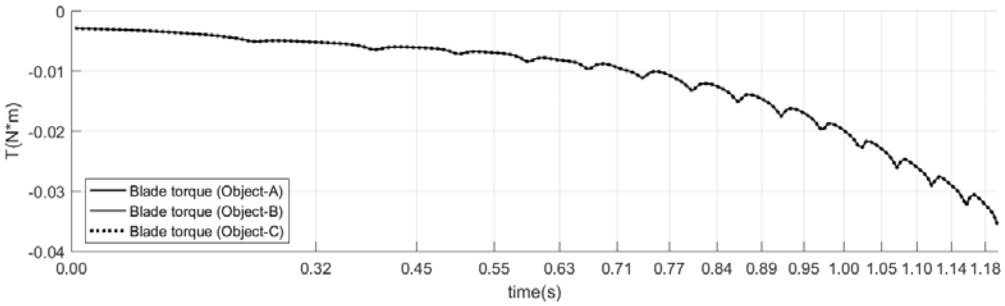


Figure 6. Vane torque

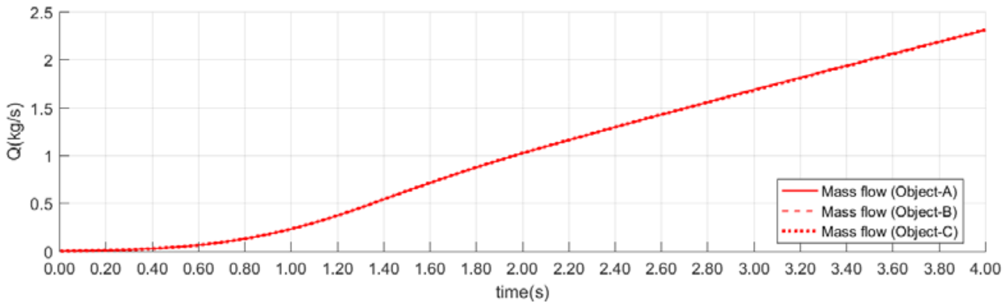


Figure 7. Mass flow

It was found by comparing the fluctuations of the fluid forces on Object-A, Object-B and Object-C (Figure 8) that the force fluctuation closely related to pump revolution; the force fluctuation had the same main frequency as the vane; with decreasing gap between the moving object and pipe wall, the force fluctuated more sharply because the differential pressure on the moving object dominated and the pressure fluctuation caused by pump revolution could be transferred by the fluid to the downstream.

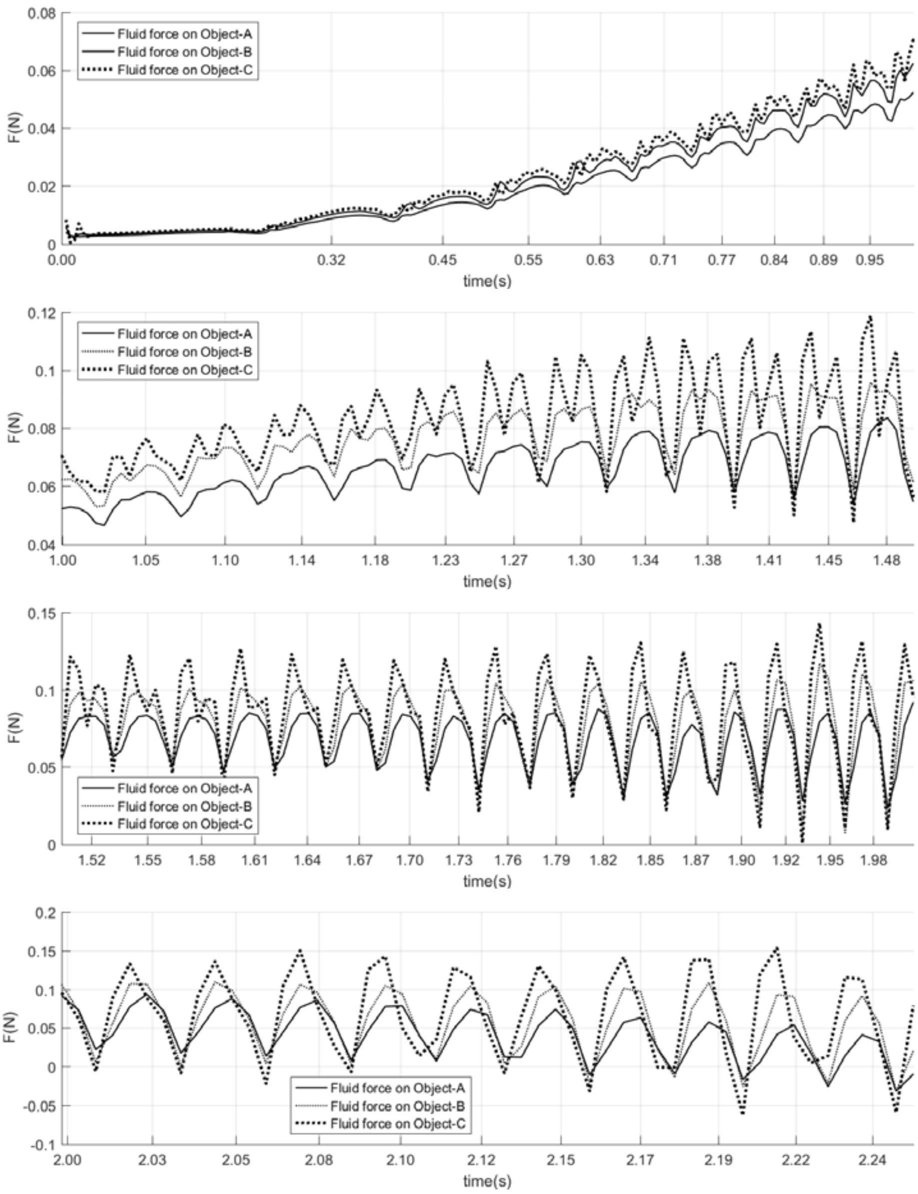


Figure 8. Fluid Force Fluctuation (Time scale as the inverse number of vane frequency)

It could be seen from comparison of fluid forces on all the objects that a law of force fluctuation existed, but the high pressure fluid from the pump flowed away from a big gap without force on the object (Figure 9), so the forces on Object-A, Object-B and Object-C were significant different and finally affected their movement processes as shown in Figure 10 and Figure 11. it could be seen that Object-C always moved to the outlet, whereas Object-A and Object-B moved in the reverse direction because the high pressure fluid flowed through the gap before a high pressure first and then a low pressure, which stopped the forward movement of the object, occurred in the front of the object (Figure 12).

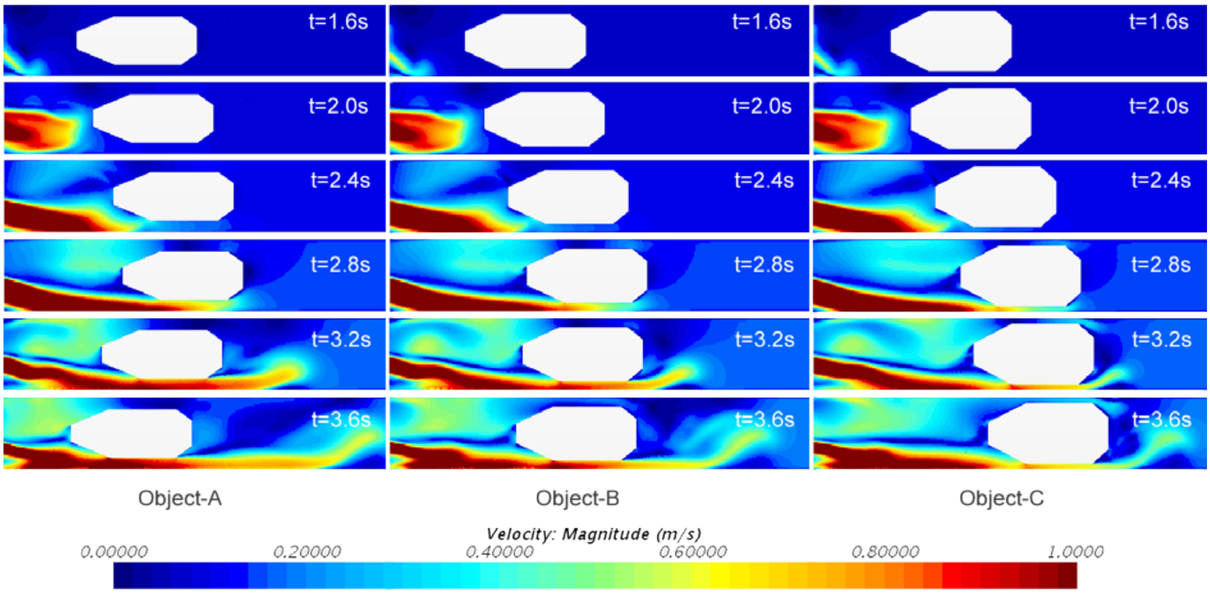


Figure 9. Velocity Distribution Cloud (z=0 m)

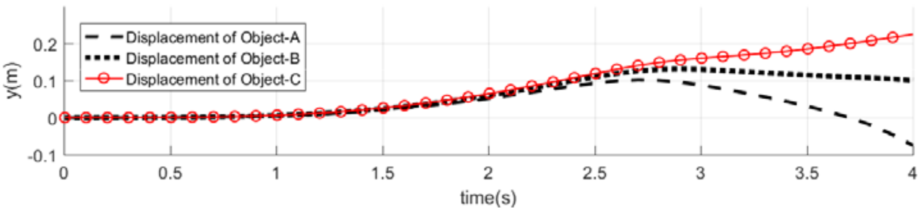


Figure 10 Moving object displacement

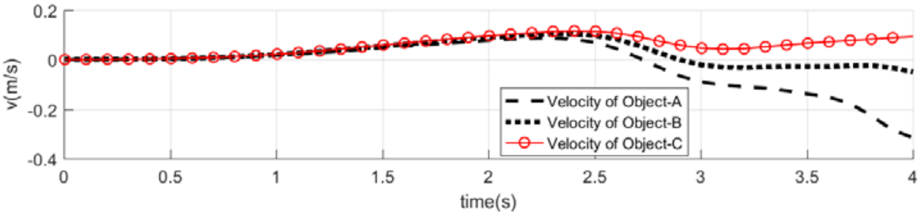


Figure 11 Velocity

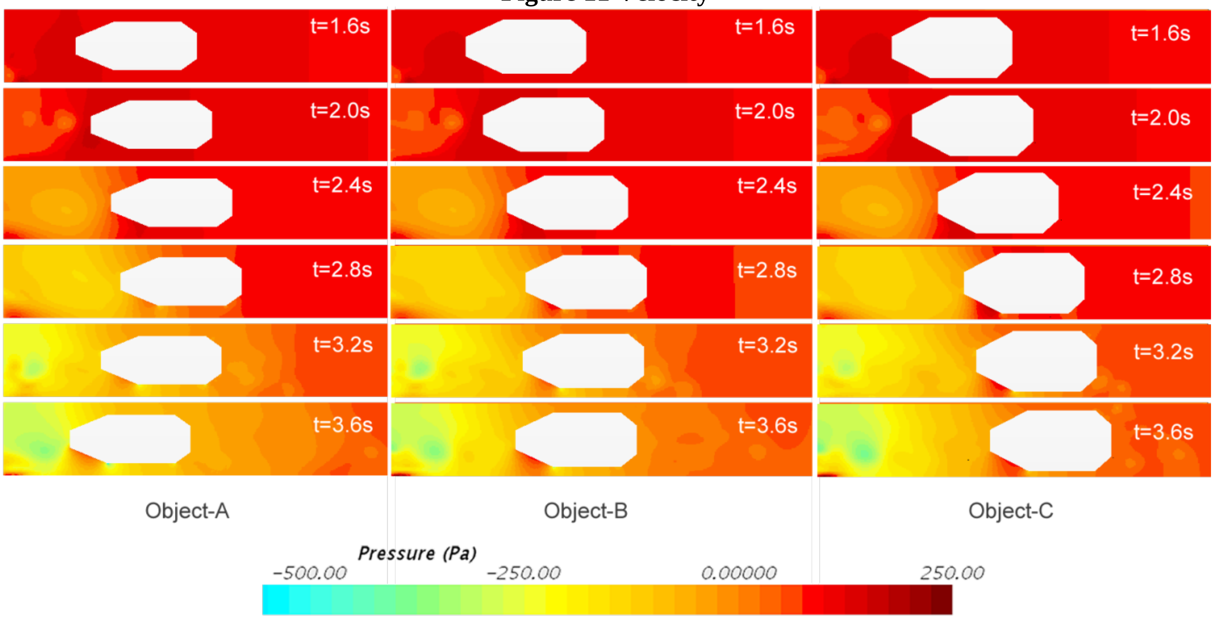


Figure 12. Pressure Distribution Cloud (z=0 m)

The pump revolution would lead to pressure fluctuation of the water flow at the pump outlet; with increasing revolution speed of the pump, both fluctuation frequency and magnitude increased, resulting in increasing force fluctuation on the object. However, the force on the object fluctuated more greatly when the gap was reduced, especially when Object-B and Object-C moved more slowly than Object-A (Fig. 7), maybe because the limited calculation capacity of the overset mesh method, which required mesh densification at the gap according to the gap dimension or improved the fluid force by further reducing the time step, leaded to impossible numerical calculation on a smaller mesh scale in this paper.

Conclusions

The overset mesh method was used for numerical calculation of the movement process of a moving object in a pipe driven by a pump, the fluctuation of the fluid force on the moving object is minimized by relevant settings, and the movement processes and laws of the moving objects with different diameters were analyzed in this paper. The following conclusions are given:

1. The fluid forces on the moving object have the same fluctuation frequency as the vane;
2. The vane had the same fluid forces and torques when the pump was driving different moving objects;
3. Object-A, Object-B, and Object-C moved in a law, but differently. In particular, Object-B and Object-C moved in the reverse direction mainly because of a pressure difference between before and after the object. This maybe did not match with the actual situation maybe because the time step used for calculation in this paper did not meet the calculation of the flow field at high vane frequency. Thus, the calculation will be improved by further mesh densification or time step reduction if the calculation condition permits.

References

[1] Allievi.The theory of water hammer.English Translation by Halmos.ASME, New York, 1925.
[2] Bergeron.Study on the steady Fluctuation in Water-filler conduits.General Graphical Solution.Revue General de L Hydraulique, 1935, 1(1).
[3] Wylie, V.L.Streeter.Fluid transients[M]. New York: McGraw-Hill, 1978
[4] Akimoto.Narumi. *Water Hammer and Pressure Fluctuation*[M]. Beijing: China Power Press, 1981.
[5] Tanaka T, Tsukamoto H.Transient Behavior of a Cavitating Centrifugal Pump at Rapid Change in Operating Conditions—Part 1: Transient Phenomena at Opening/Closure of Discharge Valve[J].Nihon Kikai Gakkai Ronbunshu B Hen/transactions of the Japan Society of Mechanical Engineers Part B, 1999, 121(4).
[6] Tanaka T, Tsukamoto H, Tanaka T, et al.Transient Behavior of a Cavitating Centrifugal Pump at Rapid Change in Operating Conditions—Part 2: Transient Phenomena at Pump Startup/Shutdown[J].Journal of Fluids Engineering, 1999, 121(4): 850-856.
[7] Tanaka T, Tsukamoto H.Transient Behavior of a Cavitating Centrifugal Pump at Rapid Change in Operating Conditions—Part 3: Classifications of Transient Phenomena[J].Nihon Kikai Gakkai Ronbunshu B Hen/transactions of the Japan Society of Mechanical Engineers Part B, 1999, 121(4): 857-865.
[8] Wu Dazhuan. THE STUDY ON TRANSIENT CHARACTERISTICS OF SPECIAL CENTRIFUGAL PUMP DURING QUICK START PROCESS, PhD Thesis, Hangzhou: *ZHEJIANG UNIVERSITY*, 2004.
[9] Wang Leqin, Wu Dazhuan, Hu Zhengyu et al. NUMERICAL SIMULATION ON STARTING CHARACTERISTIC OF VANE PUMP BY BOND GRAPH[J]. *JOURNAL OF ENGINEERING THERMOPHYSICS*, 2004, 25(3): 417-420.
[10] Wu Dazhuan, Wang Leqin, Hu Zhengyu. Numerical simulation of centrifugal pump's transient performance during rapid starting period[J]. *JOURNAL OF ZHEJIANG UNIVERSITY* (ENGINEERING SCIENCE), 2005, 39(9): 1427-1430.
[11] Li Zhihui, Wu Dazhuan, Dau Weiping et al. *2-D numerical simulation on transient flow in centrifugal pump during starting period*[J] *JOURNAL OF ENGINEERING THERMOPHYSICS*.2007.
[12] Hu F F, Ma X D, Wu D Z, et al. Transient internal characteristic study of a centrifugal pump during startup process [C]. //26th IAHR Symposium on Hydraulic Machinery and Systems, Beijing, China, August 19-23, 2012: IAHRXXVI-103.
[13] Steger JL, Dougherty FC, Benek JA. A chimera mesh scheme. Mini Symposium on Advances in Mesh, Generation, ASME, 1982
[14] Juvigny X, Canonne E, Benoit C. Multimesh algorithms for the chimera method. AIAA 2004-758, 2004
[15] Henshaw WD. Automatic mesh generation. Acta Nummerica, 1996, 5. 121 148

[16] Zhang Laiping, Deng Xiaogang, Zhang Haixin. Reviews of Moving Mesh Generation Techniques and Numerical Methods for Unsteady Flow[J]. *Advances in Mechanics*, 2010, 40(4): 424-447
[17] Shih T I-P. Overset meshes: fundamentals and practical issues. AIAA 2002-3259, 2002
[18] Koomullil R, Soni B, Singh R. A comprehensive generalized mesh system for CFD applications[J].Mathe and Computers in Simulation, 2008, 78: 605-617.
[19] Wilcdx D. Turbulence modeling for CFD[J].La Canada, CA:DCW industries,Inc,1993
[20] Menter F. Two-equation eddy-viscosity turbulence models for engineering applications[J].AIAA journal,1994,32(8):1598-1605.